Design methods of Coanda effect nozzle with two streams

Michele TRANCOSSI*,[1], Antonio DUMAS[1], Shyam Sumantha DAS[2],
Jose PASCOA[2]

*Corresponding author
*,[1] Dip. di Scienze e Metodi dell'Ingeneria, Universita di Modena e Reggio Emilia
Via Amendola 2, 42122, Reggio Emilia, Italia
mtrancossi@unimore.it
[2] C-MAST, Dep. Eng. Electromecânica, Universidade da Beira Interior,
6200-Covilhã, Portugal

Abstract: This paper continues recent research of the authors about the ACHEON Coanda effect two streams nozzle. This nozzle aims to produce an effective deflection of a propulsive jet with a correspondent deviation of the thrust vector in a 2D plane. On the basis of a previously published mathematical model, based on integral equations, it tries to produce an effective design guideline, which can be adopted for design activities of the nozzle for aeronautic propulsion. The presented model allows defining a governing method for this innovative two stream synthetic jet nozzle. The uncertainness level of the model are discussed and novel aircraft architectures based on it are presented. A CFD validation campaign is produced focusing on validating the model and the designs produced.

Key Words: Coanda effect, nozzle, model, design, controls

1. INTRODUCTION

Notwithstanding an impressive volume of studies produced during last decades, the modeling activity of Coanda effect is far from being solved, at least at engineering level. The multiplicity of the involved parameters and some difficulties related to the synthesis of an effective model due to the complexity, which relates to the involved parameters.

This paper evolves the preceding research activity of the authors about the ACHEON EU FP7 project [1, 2]. This project investigates the feasibility of a novel Coanda effect [3, 4] nozzle based on two jets, which mix and adhere to a Coanda surface which is posed in the same side of the jet with higher momentum because the presence of a pressure gradient inside the synthetic jet. The project aims to verify the feasibility of the nozzle in the field of aeronautic propulsion even with unconventional aircraft architectures.

The basic architecture of the Homer nozzle, which is the main component of the system, has represented in Fig. 1, which shows the dual jet architecture and explains the behavior of the nozzle. The jet with higher momentum is represented in red and the figure shows that it attracts because of lower pressure the other jet leading Coanda adhesion, as explained in the Patent [5, 6], from which the ACHEON project has derived.

A wide range of preliminary activities has been produced. After the initial bibliographic analysis by Trancossi [7, 8] and Pascoa [9], it has been produced a combined CFD and theoretical activity. A preliminary model of the Coanda adhesion of a fluid as a function of angular velocity of a two ducted fan units [10] has realized. Turbulence has been

investigated [11] and a preliminary analysis of the Coanda adhesion based on a simplified exact integration of Navier-Stokes equation under a set of simplified hypothesis [12] has produced. After these results, it has been possible to obtain an effective model of the nozzle using Bernoulli theorem integral equations and the inviscid irrotational flow hypothesis by Bradshaw [13]. In particular, it takes advantage of the model defined by Keen [14, 15], which allow to write a system integral equation, even if approximate. It has been clearly demonstrated that the presence of two streams influences the adhesion of the synthetic fluid stream on two Coanda surfaces, so that the synthetic stream tends to adhere to the surface on the side of the jet with higher momentum.
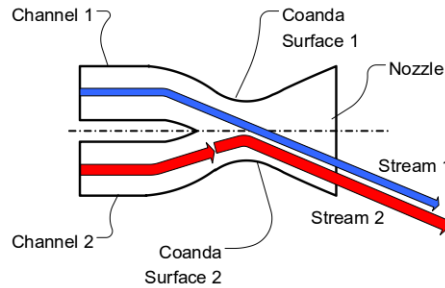

Fig. 1 - Graphical description of the Homer nozzle

In this paper it has been introduced a novel procedure based on constructal law [14-16] to define an adaptive control synthesis together with an effective design methodology. A characterization of the geometrical and physical parameters and an analysis of their influence have been obtained. Constructal optimization will allow to define a novel simplified architecture that has been defined by the choking of a fluid jet coming from only one rotor.
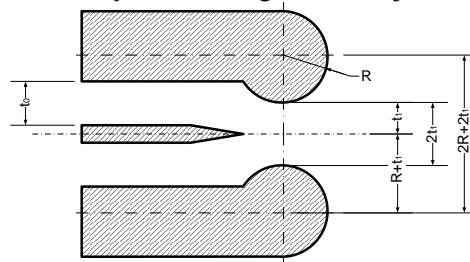

Fig. 2 - Nozzle architecture indicating physical and geometric parameters

2. REFERENCES ON COANDA ADHESION

In a previous work, the two-stream nozzle has been modeled by considering two different and independent stream sources [13]. A detailed schema is presented in Fig. 2, showing principal geometric and fluid-dynamic magnitudes, which are involved in the definition of the system. According to the schema in Fig. 2, it is possible to produce an effective functional schematization of the system, which has presented in Fig. 3 and that considers general system symmetry. An integral solution of the flow of a jet over a Coanda surface presents some problems of analysis. It could be possible an approximation of the surface pressures induced on the curved surface wet by the jet by analyzing a convex surface with a non-uniform external flow field. Roberts [14] has demonstrated that this analysis presents problems concerning mathematical analysis. This paper tries to define an integral model, which could simplify the system modeling and the expression of the necessary control laws

that are necessary to demonstrate the suitability of the system for any industrial application and, in particular, to aeronautic propulsion.
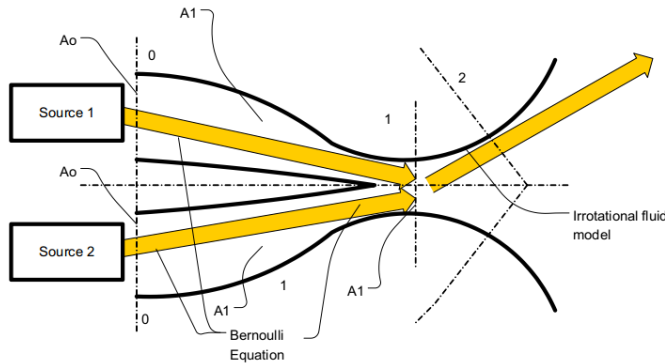

Fig. 3 - Functional schema of the nozzle

According to the schema in Fig. 3 it is possible to reassume the main equations that govern the system considering a straight jet source in two-dimensional case. It has been assumed that air is in the range of Mach number in which it can be considered incompressible (*Ma < 0.3*), even if it can be reasonably extended up to Mach 0.5. It has also assumed that the irrotational fluid-dynamic model can be applied [15, 16].

An extensive review by Bradshaw [17, 18] presented models of two-dimensional curved surfaces, from 1930, and remain the most important reference for the specific argument. In particular, Bradshaw has proposed a model of adhesion based on Bernoulli theorem

$$\theta = f \cdot \sqrt{a \cdot b \cdot \frac{P_{wall} - P_\infty}{\rho \cdot u^2}} \tag{1}$$

which applies to the adhesion of a single jet with velocity *u* and thickness *b* on a curved surface of radius *a*.

The coefficient *f* is a proportionality coefficient and includes the physical effects that take place into the adhesion of a fluid jet on a curved surface: centrifugal effect due to rotation and the friction effect on the curved surface.

This model works properly but presents a large degree of uncertainness which has been evidenced by Gibson [19] and Spalart [20]. Meroney and Bradshaw [21] have demonstrated that the growth rate on the convex wall was only half that of plane flow and that skin friction coefficients were reduced by about 10%.

Gibson [22] confirms that the turbulence in shear layers is highly sensitive to streamline curvature in the plane of the mean shear. In particular, he has confirmed that turbulent stresses and intensities are reduced by curvature when the angular momentum of the mean flow increases in the direction of the radius of curvature, as in the boundary layer on a convex wall, and increased when the angular momentum decreases with increasing radius of curvature, as in concave wall flow.

Roberts [23] has produced an analytic model of the turbulent boundary layer during Coanda adhesion on convex surface, comparing the results with the ones on a flat planes.

Newman [24] describes the flow of an incompressible jet over a surface of constant radius, assuming inviscid and non turbulent flow. Under this hypothesis the flow never separate from the surface and the radial momentum equation can be integrated across the jet to give a non-dimensional surface pressure.

$$u_p = \frac{P_s - P_a}{P_0 - P_a} = -\left[2\frac{b}{a} + \left(\frac{b}{a}\right)^2\right] \tag{2}$$

Wilchen [25], So and Mellor [27] have calculated local longitudinal velocity. They have obtained the following general simplified expression from the Bernoulli equation under the assumption $\partial P/\partial y = k\rho u^2$:

$$u/u_{pw} = \left[(P_t - P_r)/(P_r - P_{sw}) + e^{-2ky}\right]^{1/2} \tag{3}$$

where $u_{pw} = [2P_r - P_{sw}]^{0.5}$, P_r is the reference total pressure in potential core region; P_t is local total pressure; P_{sw} is static pressure at wall; and k is the inverse of the radius of curvature.

The boundary layer presents more problem of representation, especially in the present configuration with a curved surface. Assuming the irrotational fluid velocity it can be evaluated the turbulent profile of the boundary layer on a flat plane. For any generic boundary layer Von Karman (cited through Massey [29]) assumed that the velocity in the boundary layer can be expressed by a polynomial expression. For a fluid flow with a centrifugal pressure gradient it can be written the derived condition that is:

$$\mu\frac{d^2u}{dy^2} = \frac{dp}{dx} \tag{4}$$

Schlichting [30] testifies the importance of the radial pressure component and proposes a fourth order polynomial expression of the velocity into the boundary layer:

$$\frac{u(y)}{u_{max}} = 2\eta - 2\eta^3 + \eta^4 + \frac{\Lambda}{6}\eta(1-\eta)^3 \quad \text{with}\, \eta = \frac{y}{\delta} \tag{5}$$

where the shape factor due to curvature Λ is

$$\Lambda = \frac{\delta^2}{\nu} \cdot \frac{du}{dx} \tag{6}$$

The model presented by Schlichting is very interesting because allow to express the boundary layer as the composition of two different profiles, one caused by viscous effects and one caused by curvature.

Assuming that initial boundary layer thickness is negligible it is possible to assume $\Lambda = 0$ at the flux outlet 1. It can be noticed that $\Lambda > 0$ implies pressure decrease and $\Lambda < 0$ implies pressure increase and that the described profile presents a point of inflection.

The main applicative problem of the shape factor coefficient is that it varies locally depending on variable coefficients such as δ and du/dx.

3. DEFINITION OF THE PROBLEM

Considering the dynamic equilibrium of an elemental volume of fluid in rotation over the Coanda surface, it is possible to express the four concurrent effects, which applies on the fluid jet: the difference of pressure due to velocity variation at the inlet, the centrifugal force, the friction effects, interaction with external environment.

They must be evaluated and described in similar forms in a form that allow to write an effective equilibrium (7).

The fluid stream is subject to a difference of pressure that equilibrates the centrifugal force which applies because of the rotation.

$$\frac{\partial P_r}{\partial r} = \rho \cdot \omega^2 \cdot r = \rho \cdot \frac{u^2}{r} \tag{7}$$

This term is always greater than zero. It indicates the existence of a pressure gradient in the direction normal to the lines of flow and directed towards the exterior. This means that the pressure increases away from the center of curvature thus pushing the fluid toward it.

It is also subject to frictional effects that are depending on the velocity u

$$\tau = \mu \cdot \frac{\partial u}{\partial r} = c_f \cdot \frac{\rho \cdot u^2}{2} \tag{8}$$

On the basis of existing bibliography it can be possible to conclude that two fluid problems has never been analyzed except by Keen [15, 16]. The flow with higher velocity jet over the surface generates the adhesion, while the slower external one increases the complexity and could produce discrepancies into the analysis. It is possible to approximate the surface pressures induced on the curved surface wetted by the higher velocity jet, but it is not possible not possible analytically. It is then necessary to use approximate models, such as the one used by Keen [15, 16] and Trancossi [13].

The basic hypothesis of this simplified method is to represent the circular streamlines during Coanda adhesion by potential flow. It allows modeling the adhesion since that flow field has circular streamlines and satisfies the boundary condition on the circular surface. A typical wall-jet velocity profile is represented in Fig. 4.

It means that the boundary layer is thin and has reduced effects on the surface pressure (pressure can be assumed nearly constant across the boundary layer), according to Bradshaw [17] and Davenport [28], who has clearly demonstrated that circular streamline theory can provide reasonable estimates of the surface pressure, even if it can be corrected by including a small amount of work.
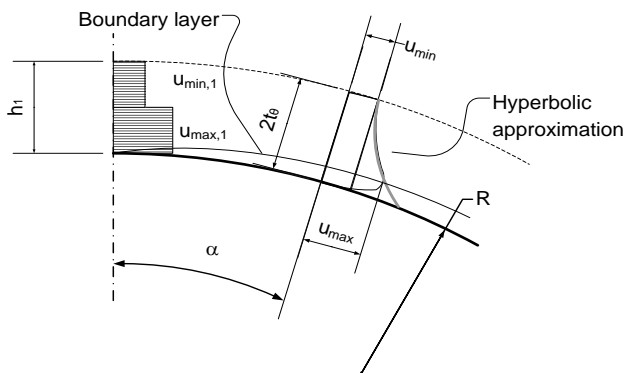

Fig. 4 - Velocity profiles

Some further discussions are necessary about the velocity distribution. The most interesting model for describing the fluid over a circular convex surface is to assume a free stream hyperbolic profile of velocity. It means to consider a velocity distribution that approximate a segment of hyperbole. Assuming that the two jets have different velocities it can be possible to assume the velocity profile as the composition of two profiles: if u_1 is the velocity of the slower jet the velocity profile during adhesion can be described adding an

hyperbolic profile to this constant velocity a constant velocity profile u_1 in order to respect the principle of conservation of mass.

The velocity of the synthetic jet has an hyperbolic profile represented in Fig. 4, which can be defined by:

$$u(r) - u_{\min} = \frac{R}{r} \cdot \Delta u_1 \rightarrow u(r) = \frac{R}{r} \cdot \Delta u_1 + u_{\min,1} \tag{9}$$

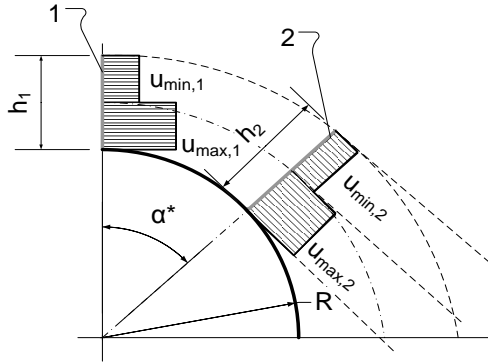where $r = R + y > R$ and $\Delta u_1 = u_{max,1} - u_{min,1}$.

Fig. 5 - Inlet and detachment conditions

It can be possible to express the system behaviour according to Fig. 5. It is important to analyze the behaviour of the fluid stream. Assuming α^* the angle of detachment, h_1 the initial thickness of the system of the jet and h_2 the thickness in correspondence of the detachment. It can be written the equation of continuity

$$u_{av,1} \cdot t_1 = \frac{u_{\max,1} + u_{\min,1}}{2} \cdot t_1 = u_{av,2} \cdot t_2 = \frac{u_{\max,2} + u_{\min,2}}{2} \cdot t_2 \tag{10}$$

The friction with the wall depends on maximum velocity because of the velocity shape. The boundary layer thickness can be then approximately calculated by equation (13) and (14):
1.   Friction term

$$h_f \cong \delta \cong c \cdot \left(\frac{\nu \cdot L}{u_{\max}} \right)^{1/n} \tag{11}$$

2.   Centrifugal term

$$h_c = \frac{u_{av}^2}{r} \frac{L^2}{u_{\max}^2} = \frac{u_{av}^2}{u_{\max}^2} (R + 0.5h)\alpha^2 \tag{12}$$

where c is a proportionality coefficient depending on the boundary layer shape.

Combining them it results:

$$\Delta h = h - h_1 = \delta_f + a_r \frac{t^2}{2} \cong \delta_f + \frac{u_{av}^2}{r} \frac{L^2}{u^2} = \delta_f + (R + 0.5h)\alpha^2 \tag{13}$$

This expression, even if different is coherent with the analysis Schlichting [30], which defines the boundary layer of the fluid over a curved surface with two different terms.
At the detachment it becomes:

$$\Delta h_{1,2} = h_2 - h_1 = h_{f_{1,2}} + h_{c_{1,2}} \cong \delta_f^* + \frac{u_{av}^2}{u_{max}^2}\left(R + 0.5 \cdot h_1\right) \cdot \alpha^{*2} \tag{14}$$

It is then possible to write the equation of conservation of energy between inlet and detachment point on a generic streamline

$$\rho \cdot u_{av,1}^2 + p_1 = \rho \cdot u_{av,2}^2 + p_2 \tag{15}$$

and the equation of conservation of mass

$$u_{av,1} \cdot h_1 = u_{av,2} \cdot \left(h_1 + h_{f_{1,2}} + h_{c_{1,2}}\right) \tag{16}$$

This relation is confirmed by CFD simulations at least in qualitative way, because they clearly shows the jet enlargement.

4. BOUNDARY LAYER MODEL

Assuming an high Reynolds number flow it can be possible to define the boundary layer on a flat plane by a simpler model than the one presented by Schlichting. A very good experimental correspondence has ensured by the following equation:

$$u = u_{max,1} \cdot \left(\frac{y}{\delta}\right)^{1/7} \tag{17}$$

which allows calculating

$$\frac{\delta}{x} = 0.166 \cdot \mathrm{Re}_x^{-1/7} \text{ and } c_f = 0.0277\mathrm{Re}_x^{-1/7} . \tag{18}$$

It is then possible to write the Bernoulli theorem in radial direction at the outlet considering one point on the wall and one on the virtual streamline that separates the two streams. It results also the value of the wall shear stress:

$$c_f = \frac{2 \cdot \tau_w}{\rho \cdot u_{max,1}^2} = 0.0277\mathrm{Re}_x^{-1/7} \rightarrow \tau_w = 0.01385 \cdot \mathrm{Re}_x^{-1/7} \cdot \rho \cdot u_{max,1}^2 \tag{19}$$

Wall shear stress is a very important parameter to be evaluated because the calculation of the fluid by integral equation will require a good representation of friction effects. It is possible to express a simplified expression of wall shear stress which consider an approximate expression of the term *du/dr*.

$$\tau_w = \mu \cdot \frac{du}{dr}\bigg|_{r=R} \cong \mu \cdot \frac{u_{max,1}}{\delta} \tag{20}$$

It can be evaluated the wall shear velocity u_τ that is:

$$u_\tau = \sqrt{\frac{\tau_w}{\rho}} \cong \sqrt{\frac{\mu \cdot u_{max,1}}{\rho \cdot \delta}} \tag{21}$$

Wall shear velocity is very important because it indicates the fluid velocity that is loss because of the wall shear stress.

On the basis of equation (18) it can be expressed an effective equation for δ:

$$\delta = 0.166 \cdot x \cdot \mathrm{Re}_x^{-1/7} = 0.166 \cdot \frac{v^{1/7}}{u_{max,1}^{1/7}} \cdot x^{6/7} = 0.166 \cdot \frac{v^{1/7}}{u_{max,1}^{1/7}} \cdot \left(R\alpha\right)^{6/7} \tag{22}$$

5. MATHEMATICAL MODEL

Equation (17) and (18) allow writing the model of the fluid during adhesion arc. Separation condition can be expressed in terms of pressure, assuming that the pressure on the curved wall is equal to the external reference pressure. In the case of a single jet the reference pressure is assumed as the pressure of the external environment. In this case, because of the presence of two jet and being adhesion strictly related to the condition of the two jets it can be assumed that the reference pressure correspond to the one of the slower jet which is always external.

Expressing $\Delta h_{1,2}$ from equation (14), the equation of conservation of mass results

$$u_{av,2} = u_{av,1} \bigg/ \left(1 + \frac{\delta_2}{h_1} + \frac{u_{av}^2}{u_{max}^2} \cdot \frac{R + 0.5 \cdot h_1}{h_1} \cdot \alpha^{*2}\right) \tag{23}$$

It is then possible to write the equation of conservation of energy between inlet and detachment point on a generic streamline

$$\rho \cdot u_{av,1}^2 + p_{av,1} = \rho \cdot \left[u_{av,1} \bigg/ \left(1 + \frac{\delta_2}{h_1} + \frac{u_{av}^2}{u_{max}^2} \cdot \frac{R + 0.5 \cdot h_1}{h_1} \cdot \alpha^{*2}\right) \right]^2 + p_{av,2} \tag{24}$$

where δ^2 is expressed by equation (22) written for $\alpha = \alpha^*$:

$$\delta_2 = 0.166 \cdot \frac{v^{1/7}}{u_{max,1}^{1/7}} \cdot \left(R\alpha^*\right)^{6/7} \tag{24'}$$

The adhesion angle can be obtained by solving equation (24) and substituting in it equation (22). The calculation has been performed by Matlab and continuos function has been obtained point by point considering that the pressure near the wall is equal to the far field pressure at detachment.

6. CFD SIMULATIONS

The above presented mathematical formulation has been deeply tested and verified by a large CFD campaign in 2D. It demonstrates that even a very elemental model such as the one proposed can work properly and can produce an effective system optimization, thus it has been produced an effective optimization of the nozzle applying constructal law.

Table 1 - Data for CFD simulations

	Units	min	Max
$u_{1,0}$	*m/s*	*0*	*30*
$u_{2,0}$	*m/s*	*30*	*60*
$u_{av,0}$	*m/s*	*30*	*30*
r	*m*	*0.028*	*0.028*
R	*m*	*0.075*	*0.15*
t	*m*	*0.028*	*0.056*

Then results will be compared and validated by 2D simulations by CFD. Confirming the results of Subhash and Dumas [11], who have analyzed a similar model, it has been verified that better results can be found by Spalart-Almaras and K-ω turbulence models, which present very similar results. K-ε model presents instead a large difference prefiguring much higher values than the ones. Calculation has been performed on a PC equipped by AMD A8-5550 quad-core processor with a clock frequency of 2.1 GHz and overclock possibility up to 3.1 GHz. The dataset in Table 1 has been assumed for calculation

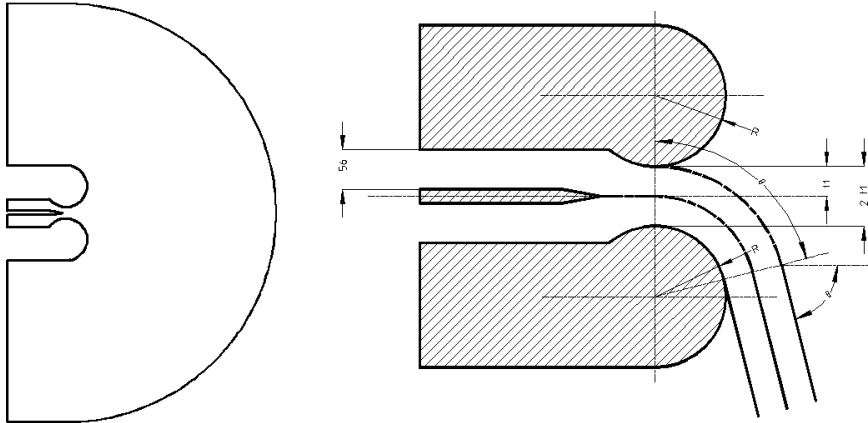A sample of the tested domains is represented for calculation in Fig. 6.

Fig. 6 - Sample of the domain and indication of main geometrical parameters

It has been performed a mesh independence analysis by a successive step mesh adaptation model. It has been finally reached a stability with a boundary layer
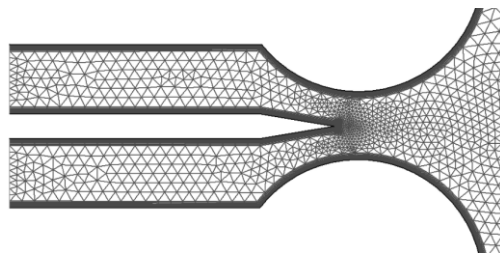

Fig. 7 - Mesh representation showing the boundary layer and the internal mesh development in the nozzle
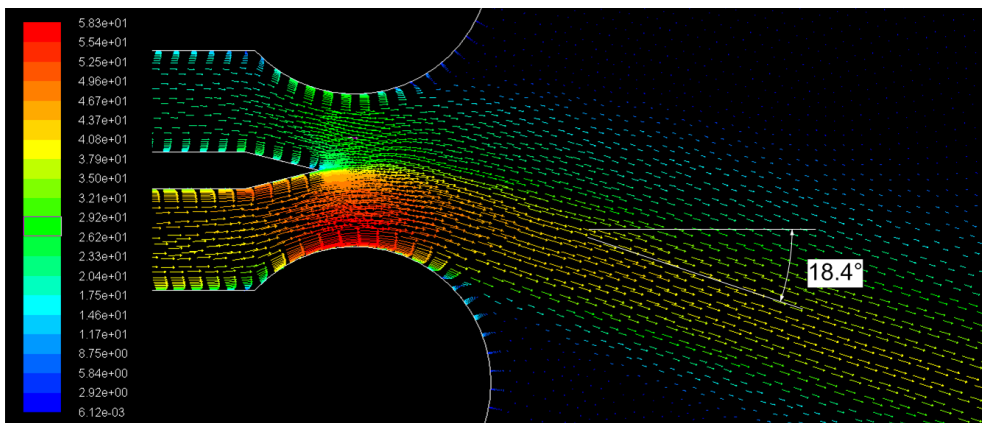

Fig. 8 - Sample of CFD simulations by Ansys Fluent 6.3 Unix Version

A visual sample of CFD simulations has been reported in Fig. 8. The mesh has been refined adopting a step by step procedure for all the proposed models.

Starting from a very coarse mesh it has been refined up the moment when the simulations on two successive meshes present errors lower than 10^{-3} in terms of velocity at the outlet.

The graph in Fig. 8 shows clearly that the obtained correlation is working in satisfactory way, with a sufficient precision.
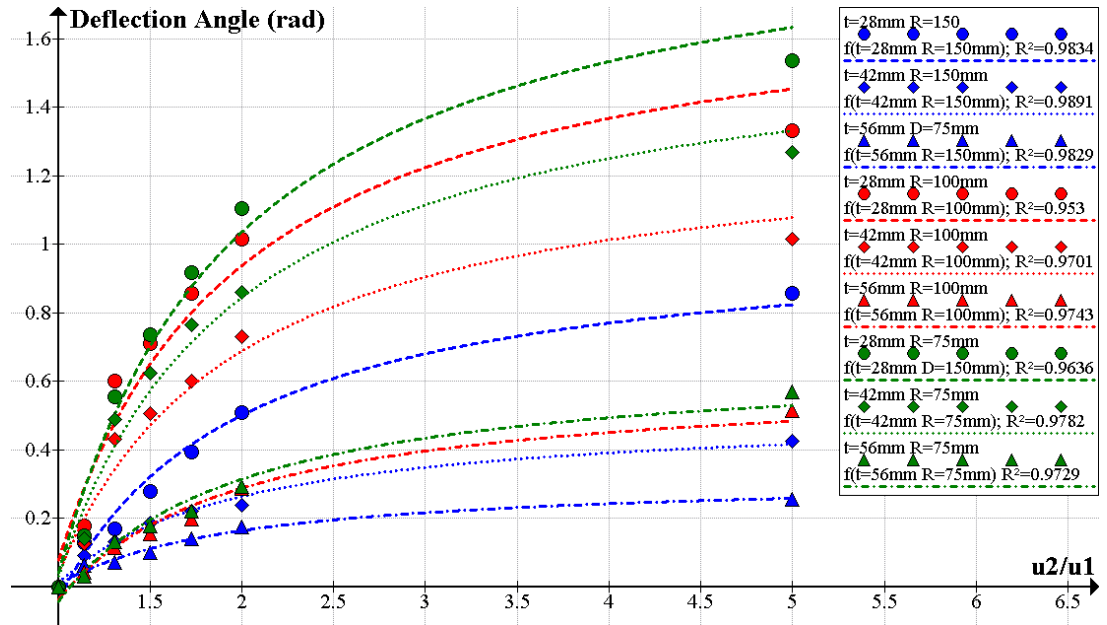

Fig. 8 - Graph of the results of CFD simulations compared with calculated results.

It is possible to verify that even if a comparison of calculated values with CFD simulations gives some errors it can be observed that most case present a satisfactory convergence.

In particular, it can be also observed that in most cases the error are certainly low and present an initial reference for predicting the performance of the system in the case of straight inlet fluid streams supplying the nozzle.

7. FURTHER CONSIDERATIONS

Das and others [33] has verified experimentally and computationally that the swirl components causes a negative effect on the Coanda adhesion. They have tested the system with inlet velocities of both 25 m/s and 15 m/s assuming a swirl velocity about 30% of axial component.

They have tested different cases and different velocities and also a non uniform velocity profile obtained by laser-Doppler experimentation on a ducted fan propeller. They have obtained angle of adhesion from 9° to 21° in different conditions.

They have in particular observed that the system presents different angles of adhesion in different conditions and especially that the angle of adhesion depends on the way in which the ducted fans rotate, because when they are counter rotating the angle of adhesion is consistently reduced.

This analysis has assumed as reference an effective model of swirling velocity which has been derived by Hennissen (34) assuming a reference velocity of 25 m/s. It has shown an effective dependence of the angle of adhesion on swirl.

Fig. 9 shows the effect of various combinations of non uniform inlet flow conditions on the exit angle.

It is significant to be noticed that case A refers to opposite swirl conditions, while cases C, D, E considers different hypotheses of combinations of the swirl components.
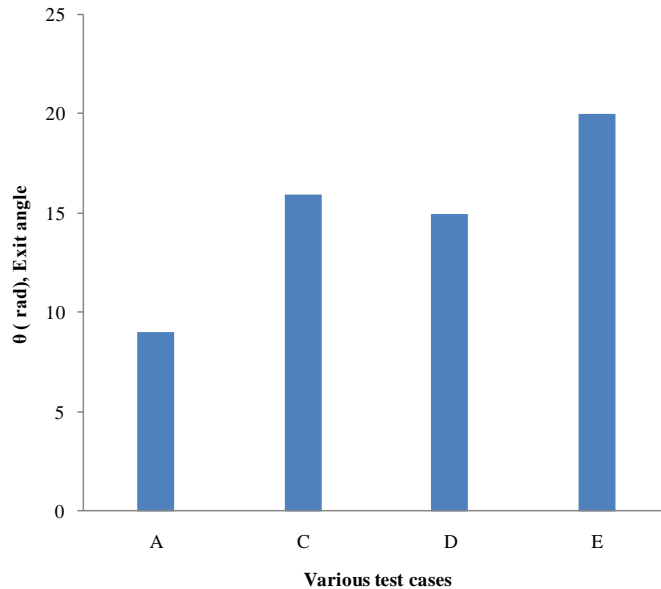

Fig. 9 - Results considering swirl velocity in different test cases

This results shows that the presence of swirl velocity reduces the jet attachment over the curved surface. This reduction of jet deflection angle may be attributed due to the generation of centripetal force inside the flow field.

The radial pressure gradient *dp/dr*, where r is the radius of the ducted fan propellers, sums to the pressure losses anticipating the detachment.

8. CONCLUSIONS

The produced results show clearly that an acceptable prediction of the performances of the nozzle can be obtained by assuming a uniform inlet. The results as shown are encouraging demonstrating the effectiveness of the calculation method proposed. It gives enough information to be used as an accurate design instrument, which is a fundamental part of the system feasibility allowing an effective prediction of performances.

It is also fundamental to assume that on the basis of the results by Das [32] it is fundamental to implement in future an adequate model for considering the effect of swirl. On the other side it will be necessary to define novel nozzle architectures which could ensure that the system can produce adequate results and can avoid swirl effect. In particular, systems, which straighten the swirl of the airflow can be adopted but also different architectures which allows to reduce the dimension of the system and can reduce the swirl components by design, can be taken into consideration.

REFERENCES

[1] *** *Aerial Coanda High Efficiency Orienting-jet Nozzle*, Research Projects, CORDIS, December 2012. Available at http://cordis.europa.eu/projects/rcn/103805_en.html.

[2] N. Dorigo, *ACHEON: thrust vectoring propulsion for future Green Aeronautics*, Cordis-Wire, January 2013. Available at https://cordis.europa.eu/wire/index.cfm?fuseaction=article.Detail&rcn=33707.

[3] H. Coanda, US Patent n. 3,261,162, *Lifting Device Coanda Effect*, 1936, USA.

[4] H. Coanda, US Patent # 2,052,869, *Device for Deflecting a Stream of Elastic Fluid Projected into an Elastic Fluid*, 1936.

[5] F. Baffigi, A. Dumas, I. Giuliani, M. Madonia, and M. Trancossi, *Ugello capace di deviare in modo dinamico e controllabile un getto sintetico senza parti meccaniche in movimento e relativo sistema di controllo*, IT patent request, RE2011A000049, 2011.

[6] F. Baffigi, A. Dumas, I. Giuliani, M. Madonia, and M. Trancossi, *Nozzle capable of deviating a synthetic jet in a dynamic and controllable manner with no moving mechanical parts and a control system thereof*, Intl. App. No.: PCT/IB2012/053198, 2012, WIPO No.: WO/2013/005132, 2013.

[7] M. Trancossi, *An Overview of Scientific and Technical Literature on Coanda Effect Applied to Nozzles*, SAE Technical Paper 2011-01-2591, 2011, doi:10.4271/2011-01-2591.

[8] M. Trancossi, and A. Dumas. *ACHEON: Aerial Coanda High Efficiency Orienting-jet Nozzle*, SAE Technical Paper 2011-01-2737, 2011, doi:10.4271/2011-01-2737.

[9] J. C. Páscoa, A. Dumas, M. Trancossi, P. Stewart, D. Vucinic, A review of thrust-vectoring in support of a V/STOL non-moving mechanical propulsion system, *Central European Journal of Engineering*, vol. **3**, Issue 3, pp 374-388, September 2013, doi: 10.2478/s13531-013-0114-9.

[10] M. Trancossi, and A. Dumas, *Coanda Synthetic Jet Deflection Apparatus and Control*. SAE Technical Paper 2011-01-2590, 2011, doi:10.4271/2011-01-2590.

[11] M. Subhash, and A. Dumas, Computational Study of Coanda Adhesion Over Curved Surface, *SAE Int. J. Aerosp.* **6**(1):260-272, 2013, doi:10.4271/2013-01-2302.

[12] M. Trancossi, Dumas, A., and D. Vucinic, , *Mathematical Modeling of Coanda Effect*, SAE Technical Paper 2013-01-2195, 2013, doi:10.4271/2013-01-2195.

[13] M. Trancossi, S. Maharshi, D. Angeli, *Mathematical modelling of a two streams Coanda effect nozzle*, ASME International Mechanical Engineering Conference and Exposition 2013, Track 1 Advances in Aerospace Technology, Topic 1.3 Advances in Propulsion System, Sand Diego November 15-21, 2013.

[14] L. Roberts, *A Theory for Turbulent Curved Wall Jets*, AIAA Paper 87-0004, 1987.

[15] E. B. Keen, *A Conceptual Design Methodology for Predicting the Aerodynamics of Upper Surface Blowing on Airfoils and Wings*, Master Thesis, Virginia Polytechnic and State University, 2004.

[16] E. B. Keen, and W. H. Mason, *A conceptual design methodology for predicting the aerodynamics of upper surface blowing on airfoils and wings*, 23rd AIAA Applied Aerodynamics Conference 6 - 9 June 2005, Toronto, Ontario Canada AIAA, 2005.

[17] P. Bradshaw, The Analogy Between Streamline Curvature and Buoyancy in Turbulent Shear Flow, *Journal of Fluid Mechanics*, Vol. **36**, Part 1, pp. 177-191, 1969.

[18] P. Bradshaw, *Effects of Streamline Curvature on Turbulent Flow*, AGARDograph, AGARDograph AG-169; 7. 1973.

[19] M. M. Gibson, W. P. Jones, B. A. Younis, Calculation of turbulent boundary layers on curved surfaces. *Phys. Fluids* **24**, 3, 386-395, 1981.

[20] P. Spalart, J. Watmuff, Experimental and numerical study of a turbulent boundary layer with pressure gradients. *J. Fluid Mech.* **249**, pp. 337-371, 1993.

[21] R. N. Meroney, P. Bradshaw, Turbulent boundary layer growth over a longitudinally curved surface. *AIAA J.* **13**, 11, pp. 1448-1453, 1975.

[22] M. M. Gibson, C. A. Verriopoulos and N. S. Vlachos, Turbulent boundary layer on a mildly curved convex surface Part 1: Mean flow and turbulence measurements, *Experiments in Fluids* **2**, 17-24, 1984.

[23] L. Roberts, *A Theory for Turbulent Curved Wall Jets*, AIAA Paper 87-0004, 1987.

[24] B. G. Newman, *The deflection of plane jets by adjacent boundaries - Coanda effect. Boundary layer theory and flow control*, p232, Ed. Zachmann, G.V., Pergamon Press, 1961.

[25] H. Wilcken, *Effect of Curved Surfaces on Turbulent Boundary Layers*, NASA TT F-11421, 1967 (translation of Turbulente Grenschichten ar Gewoelbten Ebechen, Eng. Arch., Vol. I, 1930).

[26] R. M. So, and G. L. Mellor, *An Experimental Investigation of Turbulent Boundary Layers Along Curved Surfaces*, NASA CR1940, 1972.

[27] D. A. Spence, The Lift Coefficient of a Thin, Jet-Flapped Wing, *Proceedings of the Royal Soc.*, Series A, Vol. **238**, pp. 46-68, Dec., 1956.

[28] F. J. Davenport, and D. N. Hunt, *Deflection of a Thick Jet by a Convex Surface: A Practical Problem for Powered Lift*, AIAA Paper 75-0167, 1975.

[29] B. S. Massey, *Mechanics of fluids*, 8th edition. Tylor & Francis, 2006.

[30] H. Schlichting, *Boundary-Layer Theory*, Springer, 2004, ISBN 3-540-66270-7.

[31] F. M. White, *Fluid Mechanics*, McGraw-Hill, 5th Edition, 2003.

[32] D. L. Morris, L. Hannon, and A. L. Garcia, Slip length in a dilute gas, *Physical Review* A **46** (8): 5279, 1992, doi:10.1103/PhysRevA.46.5279.

[33] S. S. Das, J. Pascoa, M. Trancossi, A. Dumas, Solving Operational Effects produced by 3D Flows on the Novel ACHEON system, *Journal of Scientific and Industrial Research*, in press.

[34] J. Hennissen, W. Temmerman, J. Berghmans and K. Allaert, *Modelling of Axial Fans for Electronic Equipment*, Flomerics Ltd., Surrey, UK, 1995.